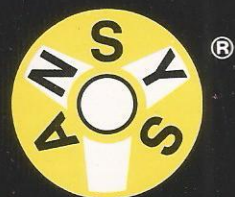
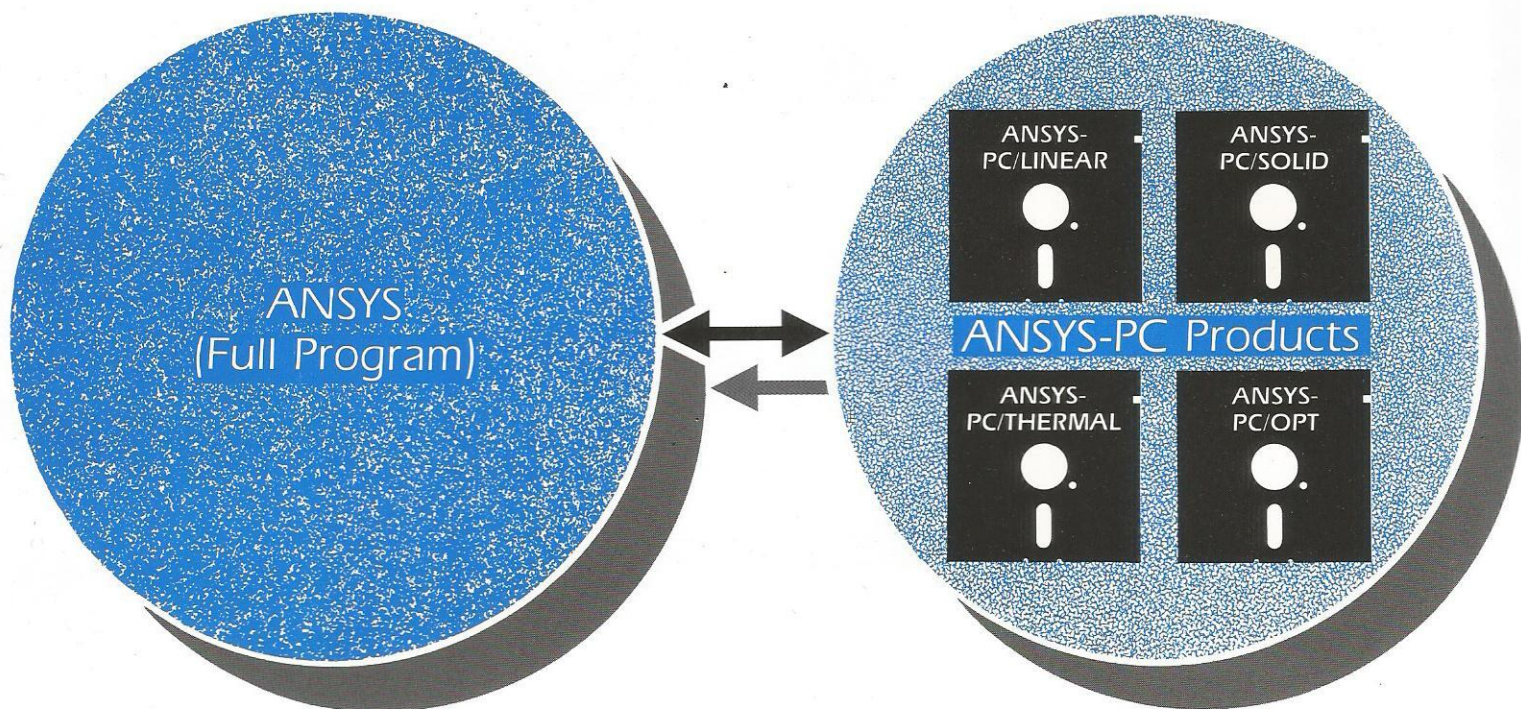


ANSYS®-PC 4.3
FAMILY OF
PRODUCTS





This flow chart symbolically illustrates the relationships between the ANSYS code and its PC subsets. Data created on the PCs may be run entirely on the PC or uploaded to a more powerful computer. Also, problem or solution datasets may be downloaded to the PC.

Output shows results from a dynamic analysis. Database manipulation routines allow sorts and selects, speeding and easing results analyses.

```

*** MASS SUMMARY BY ELEMENT TYPE ***
TYPE      MASS
1  0.165110E+01

RANGE OF ELEMENT MAXIMUM STIFFNESS IN GLOBAL COORDINATES
MAXIMUM= 0.330000E+08 AT ELEMENT 20.
MINIMUM= 0.330000E+08 AT ELEMENT 20.

*** ELEMENT STIFFNESS FORMULATION TIMES ***
TYPE  NUMBER  STIF  TOTAL CP  AVE CP
1      20      4      0.62    0.031
TIME AT END OF ELEMENT STIFFNESS FORMULATION CP= 51.300
MAXIMUM WAVE FRONT ALLOWED= 400.
EQUATION SOLUTION  ELE= 6 L.S.= 1 ITER= 1 CP= 60.56
MAXIMUM IN-CORE WAVE FRONT= 10.
MATRIX SOLUTION TIMES
READ IN ELEMENT STIFFNESSES CP= 2.660
NODAL COORD. TRANSFORMATION CP= 0.000
MATRIX TRIANGULARIZATION CP= 6.410
TIME AT END OF MATRIX TRIANGULARIZATION CP= 75.790
TIME AT START OF EIGENVALUE EXTRACTION CP= 77.800
NUMBER OF MODES AVAILABLE FROM REDUCED MATRICES= 6.
EIGENVALUE EXTRACTION TIME CP= 3.520

***** EIGENVALUE (NATURAL FREQUENCY) SOLUTION *****
MODE  FREQUENCY (CYCLES/TIME)
1      88.6119031
2      548.031603
3      1515.09195
4      2966.83938
5      4921.65692
6      8121.14936

***** REDUCED MASS DISTRIBUTION *****
ROW  NODE  DIR  VALUE
1      11  UZ  0.18533E-02
2      21  UZ  0.89662E-03
3      18  UZ  0.32508E-02
4      14  UZ  0.30547E-02
5      6   UZ  0.27972E-02
6      9   UZ  0.19892E-02

MASS(X,Y,Z) = 0.0000E+00 0.0000E+00 0.1384E-01

```

Personal computer-based engineering analysis has reached a new level of excellence with the latest versions of the ANSYS-PC family of products developed by Swanson Analysis Systems, Inc. of Houston, PA. These microcomputer-based subsets of the ANSYS finite element analysis (FEA) program can be used for a wide range of static, dynamic, and heat transfer analyses.

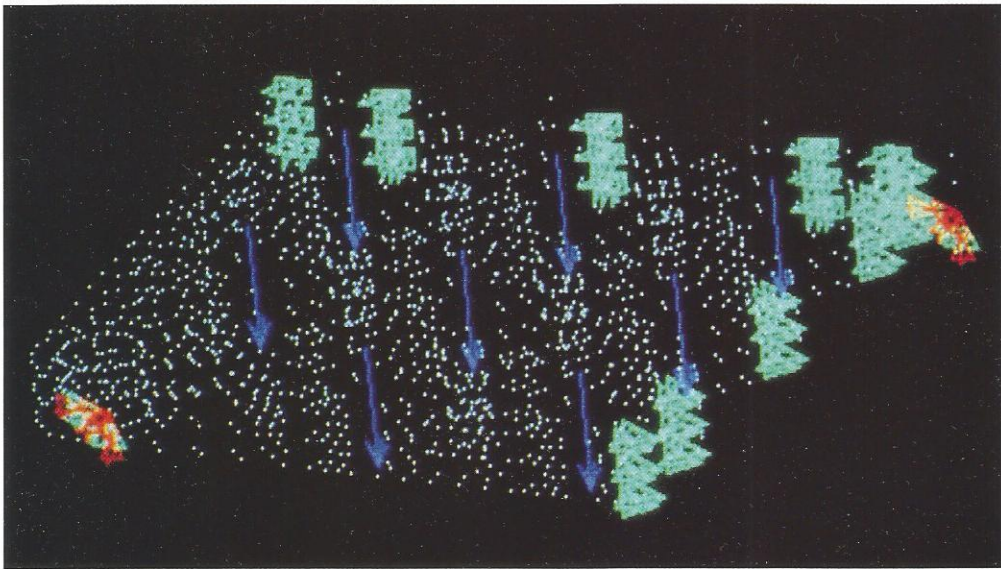
These analyses and results serve as the foundation for making accurate predictions on product performance in diverse industries ranging from aerospace, automotive and electronics to biomedical, sporting goods, and consumer products. With finite element modeling and simulation, designers can predict performance in terms of safety, reliability, and cost, without extensive prototyping.

Static and dynamic analyses are performed with ANSYS-PC/LINEAR 4.3 while thermal analyses are run with ANSYS-PC/THERMAL 4.3. Both codes are complete packages with integrated preprocessing, solution, and postprocessing. Other programs in the ANSYS-PC 4.3 family are ANSYS-PC/SOLID (for solid modeling and postprocessing), ANSYS-PC/OPT (a design optimization controller program), ANSYS-

PC/ED/LIN, and ANSYS-PC/ED/TH. The last two are educational versions of ANSYS-PC/LINEAR 4.3 and ANSYS-PC/THERMAL 4.3, respectively. These programs offer a low-cost, high-performance alternative to shared computer facilities.

Program performance also continues to improve. From the first release of the ANSYS-PC codes on 8086® processors to the latest generation of chips, the 80386®, computational speed has improved by more than an order of magnitude. The ANSYS-PC code, written in FORTRAN, has been modified with extensive assembly language coding in key areas to improve performance. For example, a large static analysis on an 80386/80387® computer ran at roughly 1.5 times the speed of a VAX™ 11/780.

Because the programs run on desktop PCs, they are readily accessible and affordable. Best of all, the ANSYS-PC family is a subset of the ANSYS program, a leader in the industry, so you don't have to learn a new command/procedure set when using more than one program. And the same high-level capabilities such as macros and user files found in the ANSYS program have also been included in the ANSYS-PC products.



Display shows boundary conditions imposed on this model of a steel casting used in a blast furnace. With 5,454 active degrees of freedom, the model ran in an hour and a half on a 16 MHz 80386 machine. Model courtesy STRUCOM, Ltd.

ANSYS-PC/LINEAR 4.3

The full-function program, ANSYS-PC/LINEAR 4.3, handles a wide range of analysis types including static, modal, dynamic response, and random vibration in one, two, and three dimensions. These analyses are performed on models built with the extensive library which contains shell, solid, beam, pipe, gap and other elements. With all these capabilities users can model and analyze large, complex systems.

The interactive preprocessor, included with the program, contains comprehensive routines for model building in two and three dimensions with node and element generation. With an extensive command set that includes pattern generation, symmetric reflections, user-controlled mesh density, and other capabilities, analysts can readily construct complex models. To help new or casual users gain familiarity with the extensive command set, an on-screen help menu is available.

ANSYS-PC/OPT 4.3

ANSYS-PC/OPT 4.3 may be used to determine a best or "optimum" solution to analyses with user-specified variables. In this iterative process, a feasible design can be developed from an initial infeasible design. Parametric design studies or "what if" scenarios can also be performed where design cost, component weight, volume, or other functions can be minimized.

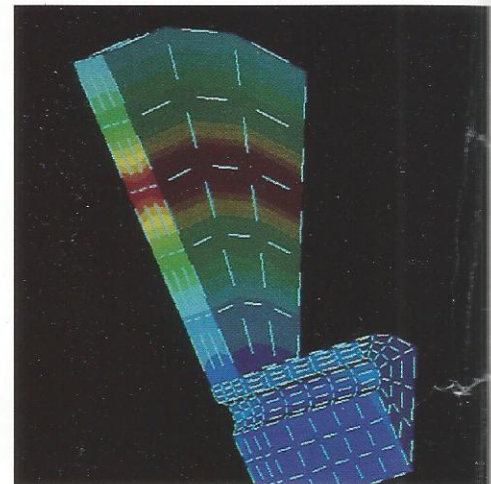
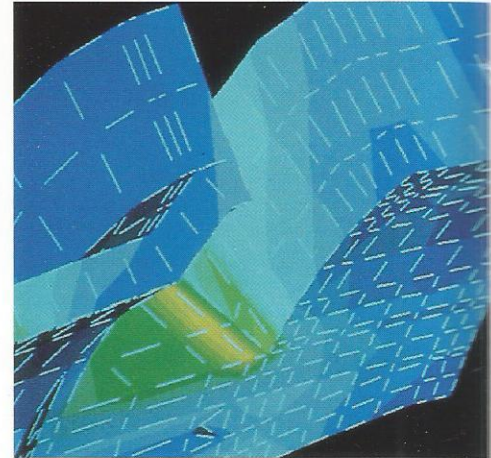
With ANSYS-PC/OPT 4.3, design aspects, such as weight, are improved without compromising the intended function. The program uses finite element analysis results to revise a computer model automatically, reducing time to rebuild complex finite element models as component geometry is

iteratively modified. Significant savings of time and money in both design and production of products can be realized. Such analyses can also confirm the safe use of a less expensive material or indicate that a reduced material volume is acceptable.

Once the finite element models are built, analysts can specify boundary conditions of acceleration, force, nodal displacement, pressure load, and thermal load. Multiple load cases can also be specified. These loads and model geometry can be visually checked with boundary condition displays and viewed from various display angles with the extensive graphics command set. Postprocessing can be done via graphic displays or with tabular listings for both static and dynamic analyses. Solution output, such as displacements, element stresses, and nodal forces, can be sorted and selected with a database language. Math operations can be used to combine or scale results. Graphic display capabilities include displaced-shape, stress contour, X-Y graphs, and shear and bending moment diagrams. Time-varying outputs such as transient dynamic response can also be displayed.

All the essential capabilities contained in the optimization module of the full ANSYS program are found in ANSYS-PC/OPT 4.3, an add-on program designed to be used in conjunction with ANSYS-PC/LINEAR 4.3 and/or ANSYS-PC/THERMAL 4.3. Also recommended for use with ANSYS-PC/OPT 4.3 is ANSYS-PC/SOLID 4.3 which can simplify meshing operations as the initial design is modified.

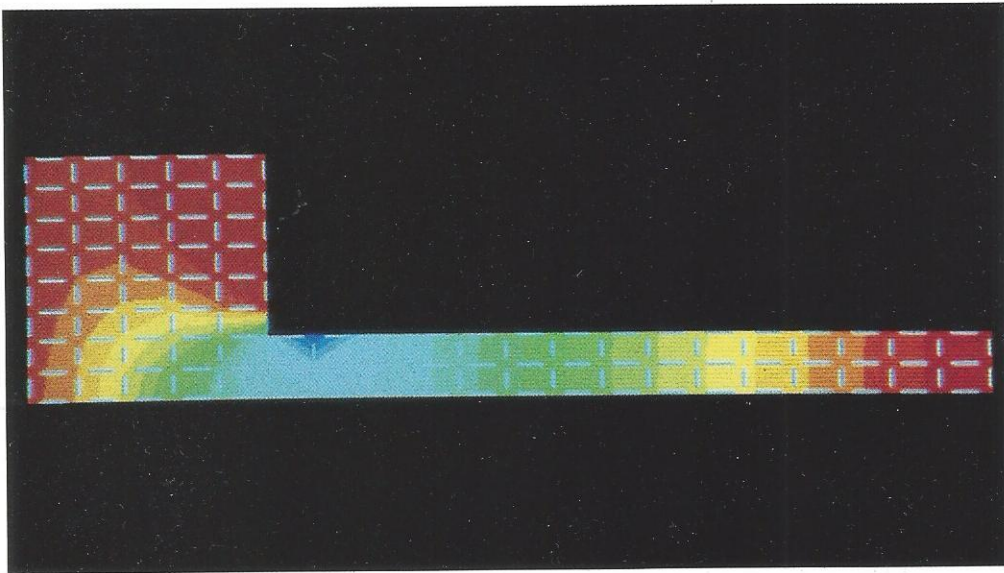
iteratively modified. Significant savings of time and money in both design and production of products can be realized. Such analyses can also confirm the safe use of a less expensive material or indicate that a reduced material volume is acceptable.



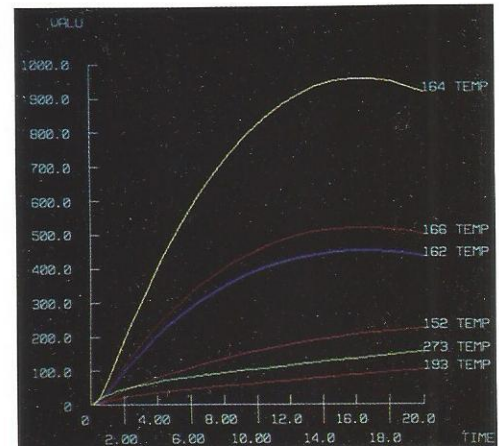
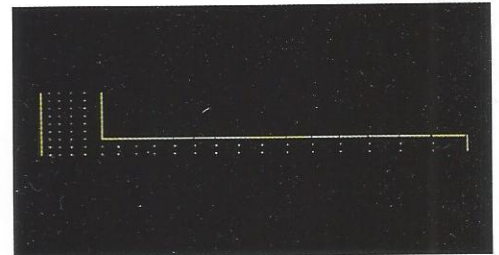
Fuel filters clean gasoline as it flows from a tank truck to an airplane. Breakage of the connector from frequent filter changes led to this new threadless design, with reliable operation checked with ANSYS-PC/LINEAR (top). Model courtesy Facet Quantek and DRD Corporation.

Loads from thermal analysis of parts such as this brake disc can be passed to ANSYS-PC/LINEAR for a coupled thermal-stress analysis, where the temperature differences yield static stress.

Call (412) 746-3304 or write Swanson Analysis Systems, Inc. for your free copy of a demo disk which highlights the capabilities of the latest generation of ANSYS-PC products.



Boundary conditions (below) on this model of a cooling fin show thermal convection loads. Results (left) show temperature-induced stress contours.



Temperature contours can be generated with most PC-based analysis codes. Few, however, can generate plots of the time-history response.

ANSYS-PC/THERMAL 4.3

Like ANSYS-PC/LINEAR 4.3, this program is fully self-contained and can be used as a stand-alone program for steady-state and transient thermal analyses. Conduction, convection, radiation, and temperature-dependent (nonlinear) material behavior can be modeled. The element library includes solids, shells, beams, and lumped thermal masses.

Analyses supported include linear/nonlinear steady-state, linear/nonlinear transient, and phase change (melting, freezing). Nonlinearities imply temperature-dependent properties, which are user-specified. For any of these analyses, multiple load steps can be specified. Automatic time-step optimization can be used for transient analyses.

Geometric preprocessing is identical to that of ANSYS-PC/LINEAR 4.3, employing the same extensive set of model generation commands. Boundary conditions can be specified in terms of temperatures, heat flows, convection surfaces, and heat generation rates. These conditions can be graphically displayed for easy verification.

The postprocessor in ANSYS-PC/THERMAL 4.3 has the same set of features as those in ANSYS-PC/LINEAR 4.3. Users can create displays of temperature contours, heat flows, time histories, and more. Data can be operated on with sorting and selecting functions to create, for example, customized reports.

DESIGN OPTIMIZATION

ANSYS design optimization employs an approximation technique which permits virtually any aspect of a design to be iteratively optimized. ANSYS-PC/OPT 4.3 can also tie ANSYS-PC/LINEAR 4.3 and ANSYS-PC/THERMAL 4.3 together to perform an optimized thermal-stress analysis as seen in the display of the finned pipe analysis.

In performing any ANSYS optimization, a user specifies design variables (problem descriptors), state variables (behavioral constraints), and the objective function (design goals). This task is eased because variables are defined with the ANSYS parametric language. A wide range of solution options are also provided. Examples of each are listed below.

Design Variables

Geometric properties of elements
(e.g., thickness, area, moment of inertia)
Geometric location of nodes
(e.g., radius, length of side, angle)

Design Variables (continued)

Location of keypoints (available within ANSYS-PC/SOLID 4.3 only)
Magnitudes or locations of loads
(e.g., pressure, heat input, etc.)
Material properties
(e.g., Young's Modulus, convection coefficient, and coefficient of thermal expansion)

State Variables

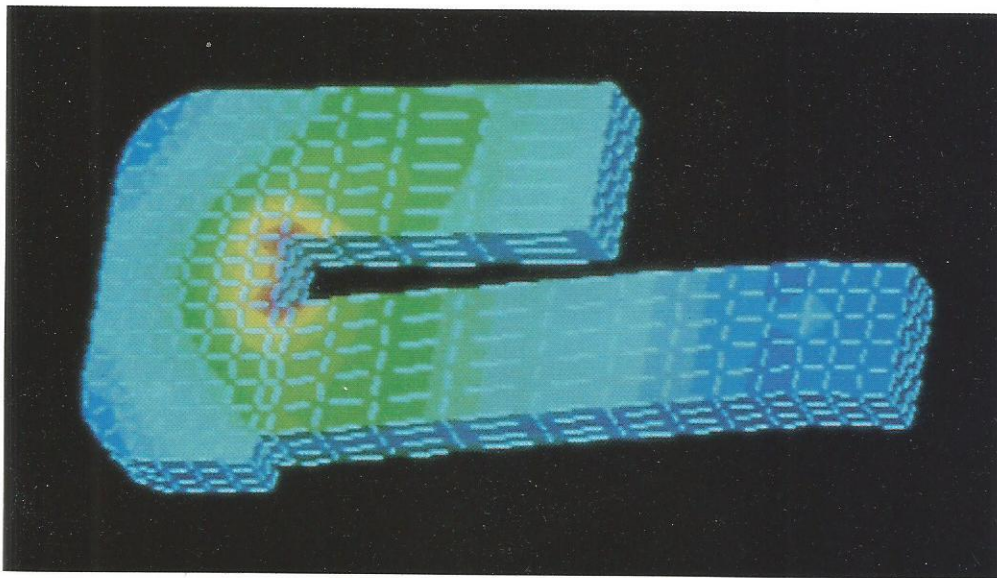
Max/min stress value (component, principal, or derived)
Max/min temperature or temperature at a point
Maximum displacement or displacement at a point
Maximum total or component strain
Natural frequency/period of vibration
Reaction force/moment summation
Total or partial heat flux
Any constraint that can be computed from analysis data (e.g., enclosed volume, factor of safety)

Objective Functions

Cost/Benefit Ratio
Deflection Limits
Stress Concentration
Total Volume, Weight
User Cost Function (can include):
Manufacturing cost
Weight budget

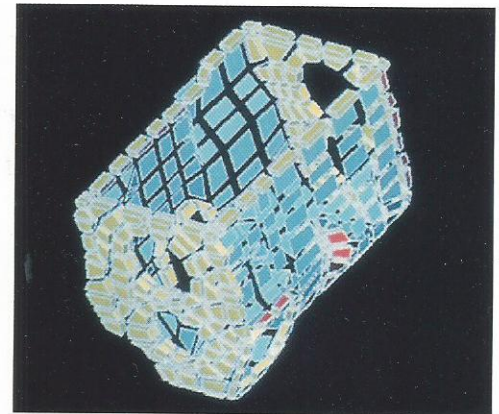
Solution

Convergence tolerances may be user controlled
Interactive or batch control of optimization looping
Up to 60 behavioral constraints can be defined
An unlimited number of design loops may be run without restarting
Results include design sensitivities



Display shows static stress in a contact pin in an electrical connector. Developed by the ANSYS distributor, Ohio CAE, the initial design was modified with ANSYS-PC/OPT to reduce static stress concentrations.

Preprocessing features such as element shrink and color coding by element type allow users to visually check geometry and ease the task of assigning boundary conditions in complex parts such as this transmission housing. *Model courtesy Tremec.*



ANSYS-PC/SOLID 4.3

The ANSYS-PC/SOLID 4.3 program includes both pre and postprocessing functions. It can prepare models for analysis with ANSYS, ANSYS-PC/LINEAR 4.3, or ANSYS-PC/THERMAL 4.3. As a postprocessor, it contains most of the full ANSYS postprocessor routines, providing a link between host-based FE solutions and PC-based results evaluation.

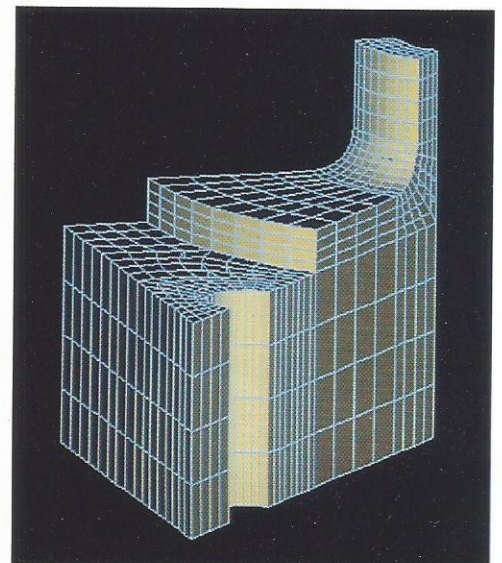
Solid modeling capabilities in the preprocessor automate one of the most time-consuming and difficult portions of finite element modeling, mesh generation. The versatility of the solid modeling routines found in the ANSYS preprocessor is duplicated almost entirely in the PC product. Cubic parametric splines are used to represent lines, surfaces, and volumes. Analysts can perform automatic intersection calculations of lines and areas, construct circular arcs, generate fillets automatically, construct tangents, and generate volumes and areas with sweeps and rotations. The automated meshing algorithms can fill both regular and

irregular areas and volumes with well-formed elements.

Models developed in this manner can be executed on the PC with PC/LINEAR or PC/THERMAL or transferred to more powerful computers running the ANSYS code. For the latter, because PC/SOLID supports all ANSYS elements, the full range of ANSYS analyses including coupled-field, fluid flow, and magnetics, can be run on PC-generated models.

The solid modeler also serves as a link to CAD systems. Through optional translators, CAD geometry from the following programs can be used as input to the ANSYS code: AutoCAD®, CADKEY™, and VersaCAD®. An IGES translator is also supported.

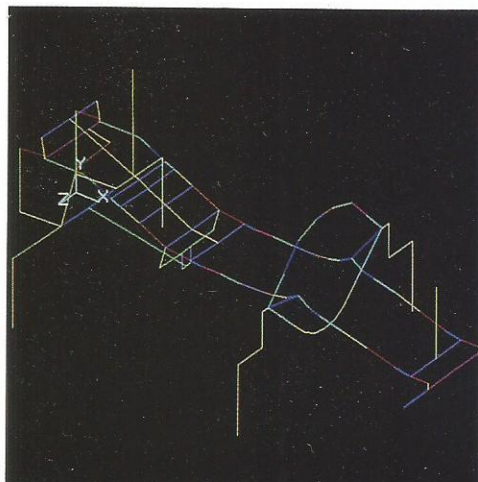
As a postprocessor, ANSYS-PC/SOLID can generate graphic displays including stress contours, displaced shape, path plots, and shear and moment diagrams. Postprocessing routines also include database search and sort capabilities.



ANSYS-PC/SOLID can rapidly generate large models such as this bearing mount with 1,216 elements.



The frame of this 170-ton WABCO hauler was modeled on a personal computer. A preliminary linear static analysis performed with ANSYS-PC/OPT indicated a possible frame weight reduction of greater than 30%. *Model courtesy Dresser Industries, WABCO Haulpak Division.*



ANSYS-PC/ED/LIN 4.3 & ANSYS-PC/ED/TH 4.3

These programs are subsets of ANSYS-PC/LINEAR 4.3 and ANSYS-PC/THERMAL 4.3, respectively. Intended for educational use, they have reduced functionality, smaller element libraries, solution sets, and a smaller problem size capacity. Because they are true subsets, the command sets and procedures are identical to the larger versions of the ANSYS code, making these educational versions excellent training tools.

MORE REASONS TO USE THE ANSYS-PC 4.3 PROGRAMS

AVAILABILITY

ANSYS-PC products may be obtained from our worldwide ANSYS Support Distributor (ASD) network as a lease or paid-up license. Continual support and upgrades are provided with leased software. Substantial discounts are available for multiple installations under one license agreement. Your local distributor has full pricing information.

QUALITY ASSURANCE

The ANSYS-PC products undergo one of the most rigorous verification procedures in the industry. Quality assurance test problems are derived from those of the full ANSYS program. This problem subset, with nearly 400 individual tests for ANSYS-PC/LINEAR 4.3 alone, fully exercises the PC products and assures high quality for ANSYS-PC program users.

SUPPORT

Product support for ANSYS-PC/LINEAR 4.3, ANSYS-PC/SOLID 4.3, ANSYS-PC/THERMAL 4.3, and ANSYS-PC/OPT 4.3 includes telephone hotline assistance from ANSYS Support Distributors with back-up from SASI. Another phase of product support is regularly scheduled training seminars which are presented by ASDs and SASI at convenient locations around the world.

ANSYS-PC code licensees (educational and paid-up license versions excluded) automatically receive all program developments and/or error corrections without additional charge. This service ensures that all program users have access to the latest improvements.

The full range of ANSYS documentation from detailed user's manuals to theoretical manuals are offered by SASI and ASDs. SASI also publishes a bimonthly newsletter, *ANSYS News*®, which is sent free of charge to all program licensees, and a semiannual applications journal, *ANSYS TechNotes*™, which is available for a one-time subscription fee.

HARDWARE REQUIREMENTS

COMPUTER:

IBM PC/XT™, PC/AT or compatibles
IBM PS/2 (all models)
COMPAQ DESKPRO 286® or 386 or compatible
Parallel printer port required
Math coprocessor (8087®, 80287® or 80387) required,
Weitek™ processor optional

MEMORY:

640KB Minimum
Add-in boards (conforming to Lotus®/Intel®/Microsoft®
Expanded Memory Specification) supported.

DISK DRIVES:

Hard: 20MB required (Larger disk recommended)
Diskette: 5.25-inch 360KB, or 3.5-inch 720KB

OPERATING SYSTEM:

PC DOS or
MS-DOS™: Revision 3.1 or later recommended
DOS 4.0 on IBM hardware

GRAPHICS:

Color
IBM CGA (standard graphics) or compatible
(resolution: 320 x 200) (Not recommended)
IBM EGA (enhanced graphics) or compatible
with 128KB or 256KB of memory
(resolution: 640 x 350)
IBM VGA (professional graphics) or compatible
(resolution: 640 x 480)
Vermont Microsystems Image Manager™ 1024
(resolution: 1024 x 800)

A variety of compatible video monitors are available for the above graphics cards.

Monochrome

Hercules™ or compatible
(resolution: 720 x 348)
AT&T 6300 WGS
(resolution: 640 x 400)

OTHER:

Microsoft mouse supported.

APPLE COMPUTER:

Macintosh II
2 Megabytes RAM (Minimum)
40 Megabyte Hard Disk (Minimum)
640 X 480, 256-Color Monitor (Required)

ANSYS and ANSYS News are registered trademarks of Swanson Analysis Systems, Inc. ANSYS TechNotes is a trademark of Swanson Analysis Systems, Inc. Apple and Macintosh are registered trademarks of Apple Computer Inc. AutoCAD is a registered trademark of Autodesk, Inc. CADKEY is a trademark of Micro Control Systems, Inc. COMPAQ and COMPAQ DESKPRO 286 and 386 are registered trademarks of COMPAQ Computer Corp. Hercules is a registered trademark of Hercules Computer Technology. IBM is a registered trademark of International Business Machines Corp. Image Manager is a trademark of Vermont Microsystems, Inc. Intel, 8086, 8087, 80287, 80386, and 80387 are registered trademarks of Intel Corp. Lotus is a registered trademark of Lotus Development Corp. MS-DOS is a trademark of Microsoft Corporation. Microsoft is a registered trademark of Microsoft Corp. PC/XT, PC/AT, and PS/2 are trademarks of International Business Machines Corp. VAX is a trademark of Digital Equipment Corp. VersaCAD is a registered trademark of Versacard Corp. Weitek 1167 is a trademark of Weitek Corporation. NEC is a trademark of Nippon Electric Corporation. 68881 is a registered trademark of Motorola, Inc.

ANSYS-PC/LINEAR 4.3 FEATURES

ELEMENT LIBRARY

SPRING
1-D, 2-D, 3-D
MASS
1-D, 2-D, 3-D
TRUSS/SPAR
2-D, 3-D
BEAM
2-D, 3-D Symmetrical
2-D, 3-D Tapered Unsymmetrical
PIPE
Straight, Elbow, Tee
SOLID
2-D 4-Node Quadrilateral
8-Node Quadrilateral
6-Node Triangle
(All have plane stress, plane strain and axisymmetric options)
3-D 8-Node Hexahedron
10-Node Tetrahedron
PLATE/SHELL
3-D, Membrane and/or Bending
4-Node Flat
Axisymmetric, Conic Shell
GAP/CONTACT
1-D, 2-D, 3-D

GENERAL MATRIX COUPLING

SOLUTION MODULES
LINEAR STATICS
GEOMETRIC NONLINEARITIES
Gap/Contact
Stress Stiffening
LINEAR DYNAMICS
Mode/Frequency Extraction
Mode Superposition
Harmonic Response
Linear Transient
PSD/Random Response
Spectrum Response

MATERIAL PROPERTIES

Isotropic
Orthotropic
Temperature-Dependent

BOUNDARY CONDITIONS

Acceleration (Gravity)
Distributed Load/Pressure
Nodal Force
Node Displacement/Rotation
Rotational Velocity/Acceleration
Temperature (Thermal Load)

ANSYS-PC/THERMAL 4.3 FEATURES

ELEMENT LIBRARY

CONDUCTING BAR
2-D, 3-D
CONVECTION LINK
2-D, 3-D
RADIATION LINK
2-D, 3-D
CONDUCTING SOLID
2-D, 3-D
Quadrilateral, Triangle, Hexahedron, Tetrahedron
Axisymmetric
PLATE/SHELL
3-D
LUMPED MASS
1-D, 2-D, 3-D

NONLINEAR

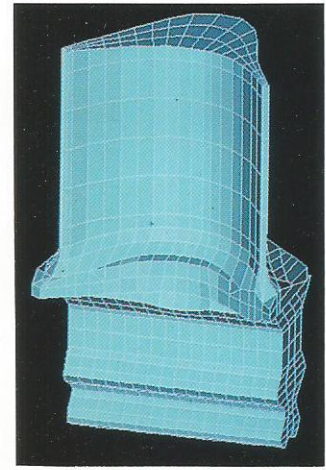
Convection
Radiation
Temperature-Dependent Properties
TRANSIENTS
Linear
Nonlinear
Convection
Phase Change
Radiation
Temperature-Dependent Properties

MATERIAL PROPERTIES

Isotropic
Orthotropic
Temperature-Dependent

BOUNDARY CONDITIONS

Convective Surface
Heat Generation Rate
Node Temperatures
Node Heat Flows



Run time for this 3-D solid model of a turbine blade was less than three hours on a 16MHz 80386 machine. Prepared by STRUCOM, Ltd., the British ANSYS Support Distributor, the model had nearly 1,500 3-D solid elements. Shown on the cover is an enlargement of the stresses at the root of this blade.

GRAPHICS, PRE AND POSTPROCESSING

Database Language
Hidden Line Removal
Light Source Display
Menu Mode
On-Line Documentation/Help
Pick Functions
Raster and Vector Mode
Rotate
Scale
Section Display
Windows
Zoom

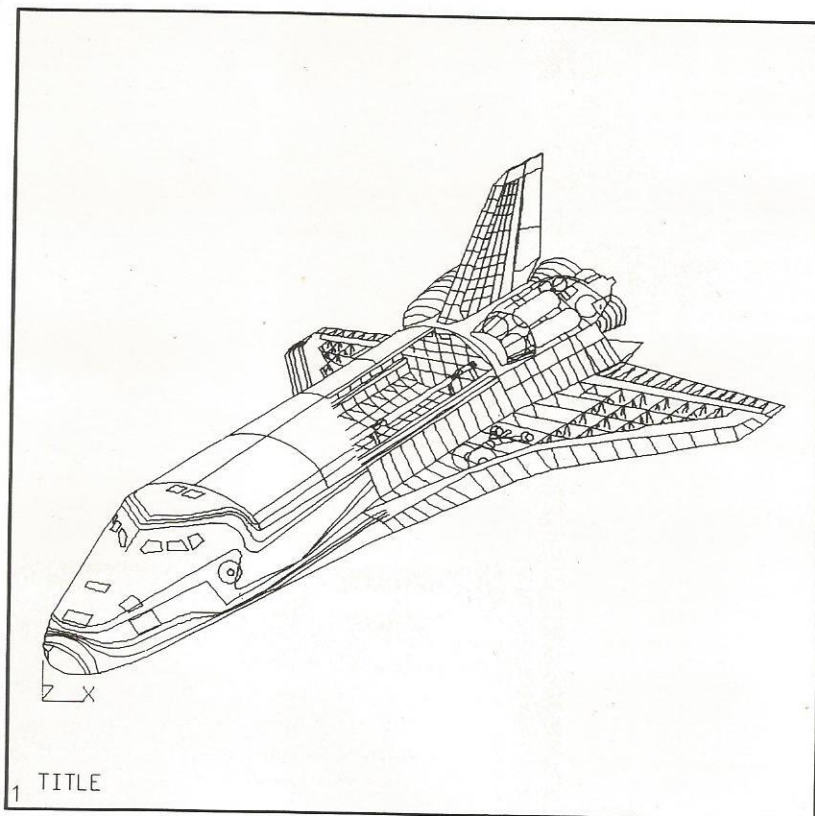
PREPROCESSING

Boundary Condition Display
Data Check
Display Functions
Macros
Mesh Generation
Node, Element (Standard)
Solid (Optional)
Neutral Files
Parametric Input
Userfiles
Wavefront Reordering

POSTPROCESSING

Animation
Combination of Results
Contour Displays
Displaced Shape Displays
Margin of Safety
Math Operations
Path Display
Section Plots
Strain, Stress
Stress Deviation
X-Y Graph Display

Data from several PC-based CAD packages can be translated into geometry for ANSYS analysis. For example, this space shuttle model was created with the AutoCAD package.

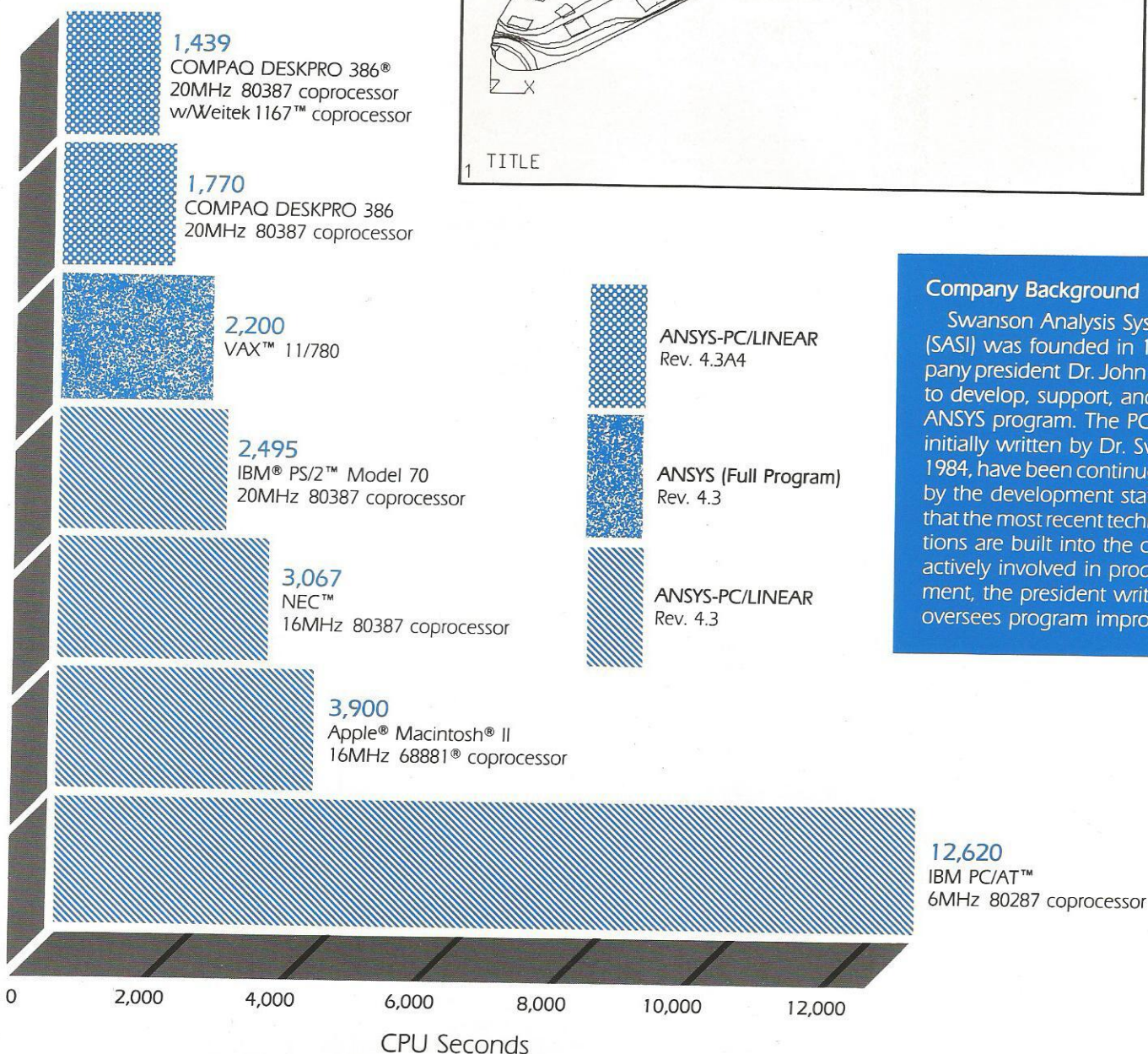


ANSYS 4.3
AUG 28 1987
15:44:01
PREP7 LINES

ZU=1
DIST=7.14
XF=6.5
YF=5.21

Benchmark Times

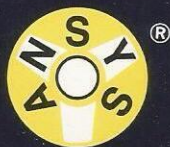
With each successive generation of hardware, the performance of the ANSYS-PC codes has improved. Since the initial release of ANSYS-PC/LINEAR in 1985, execution speed has increased by more than an order of magnitude. This improvement now brings minicomputer performance to an engineer's desktop.



Company Background

Swanson Analysis Systems, Inc. (SASI) was founded in 1970 by company president Dr. John A. Swanson to develop, support, and market the ANSYS program. The PC products, initially written by Dr. Swanson in 1984, have been continuously revised by the development staff to ensure that the most recent technical innovations are built into the codes. Still actively involved in product development, the president writes code and oversees program improvements.

NOTE: Solution times are for a model with 1,020 3-D solid elements, a maximum node number of 2,381 with 4,839 active DOF, 1 load step, a maximum wavefront of 255 and an RMS wavefront of 148.6. Data preparation (PREP) time of approximately 5% not included.



Swanson Analysis Systems, Inc.
P.O. Box 65, Johnson Road
Houston, PA 15342-0065
Phone: (412) 746-3304
TWX: 510-690-8655
FAX: (412) 746-9494